


# IPC-7525

## Stencil Design Guidelines

**IPC-7525**

May 2000

A standard developed by IPC

2215 Sanders Road, Northbrook, IL 60062-6135
Tel. 847.509.9700 Fax 847.509.9798
www.ipc.org

**The Principles of Standardization**

In May 1995 the IPC's Technical Activities Executive Committee adopted Principles of Standardization as a guiding principle of IPC's standardization efforts.

**Standards Should:**
- Show relationship to Design for Manufacturability (DFM) and Design for the Environment (DFE)
- Minimize time to market
- Contain simple (simplified) language
- Just include spec information
- Focus on end product performance
- Include a feedback system on use and problems for future improvement

**Standards Should Not:**
- Inhibit innovation
- Increase time-to-market
- Keep people out
- Increase cycle time
- Tell you how to make something
- Contain anything that cannot be defended with data

**Notice**

IPC Standards and Publications are designed to serve the public interest through eliminating misunderstandings between manufacturers and purchasers, facilitating interchangeability and improvement of products, and assisting the purchaser in selecting and obtaining with minimum delay the proper product for his particular need. Existence of such Standards and Publications shall not in any respect preclude any member or nonmember of IPC from manufacturing or selling products not conforming to such Standards and Publication, nor shall the existence of such Standards and Publications preclude their voluntary use by those other than IPC members, whether the standard is to be used either domestically or internationally.

Recommended Standards and Publications are adopted by IPC without regard to whether their adoption may involve patents on articles, materials, or processes. By such action, IPC does not assume any liability to any patent owner, nor do they assume any obligation whatever to parties adopting the Recommended Standard or Publication. Users are also wholly responsible for protecting themselves against all claims of liabilities for patent infringement.

**IPC Position Statement on Specification Revision Change**

It is the position of IPC's Technical Activities Executive Committee (TAEC) that the use and implementation of IPC publications is voluntary and is part of a relationship entered into by customer and supplier. When an IPC standard/guideline is updated and a new revision is published, it is the opinion of the TAEC that the use of the new revision as part of an existing relationship is not automatic unless required by the contract. The TAEC recommends the use of the lastest revision. Adopted October 6. 1998

**Why is there a charge for this standard?**

Your purchase of this document contributes to the ongoing development of new and updated industry standards. Standards allow manufacturers, customers, and suppliers to understand one another better. Standards allow manufacturers greater efficiencies when they can set up their processes to meet industry standards, allowing them to offer their customers lower costs.

IPC spends hundreds of thousands of dollars annually to support IPC's volunteers in the standards development process. There are many rounds of drafts sent out for review and the committees spend hundreds of hours in review and development. IPC's staff attends and participates in committee activities, typesets and circulates document drafts, and follows all necessary procedures to qualify for ANSI approval.

IPC's membership dues have been kept low in order to allow as many companies as possible to participate. Therefore, the standards revenue is necessary to complement dues revenue. The price schedule offers a 50% discount to IPC members. If your company buys IPC standards, why not take advantage of this and the many other benefits of IPC membership as well? For more information on membership in IPC, please visit www.ipc.org or call 847/790-5372.

Thank you for your continued support.




IPC-7525

# Stencil Design Guidelines

Developed by the Stencil Design Task Group (5-21e) of the Assembly and Joining Process Committee of IPC

Users of this standard are encouraged to participate in the development of future revisions.

Contact:

IPC
2215 Sanders Road
Northbrook, Illinois
60062-6135
Tel 847 509.9700
Fax 847 509.9798

# Acknowledgment

Any Standard involving a complex technology draws material from a vast number of sources. While the principal members of the Stencil Design Task Group (5-21e) of the Assembly and Joining Processes Committee are shown below, it is not possible to include all of those who assisted in the evolution of this standard. To each of them, the members of the IPC extend their gratitude.

**Assembly and Joining Processes Committee**

Chairman
James Maguire
Intel Corporation

**Stencil Design Task Group**

Chairman
William E. Coleman, Ph.D
Photo Stencil Inc.

**Technical Liaisons of the IPC Board of Directors**

Stan Plzak
Pensar Corp.

Peter Bigelow
Beaver Brook Circuits Inc.

**Stencil Design Task Group**

Kermit Aguayo, XeTel Corporation
Syed Sajid Ahmad, Micron Technology Inc.
Sherman M. Banks, Finisar, Inc.
Ricky Bennett, DEK USA Inc.
Craig P. Brown, DEK USA Inc.
Maureen A. Brown, Kester Solder Division
Magdalen Christopher, Litton Systems Canada Ltd.
Richard Clouthier, AMTX/Division of Photo Stencil
William E. Coleman, Ph.D., Photo Stencil Inc.
Jerry Cupples, Interphase Corp.
Charlie Davis, RadiSys Corporation
Ashok Dhawan, C-MAC Electronic Systems Inc.
Kantesh Doss, PhD, Nokia Mobile Phones Ltd.
Caldon Driscoll, EMPF/ACI
Frank V. Grano, SCI Systems Inc. Government Division
Ryan Grant, MCMS
Andy Hilbert, Plexus Corp./ Electronic Assembly
Jay B. Hinerman, DEK USA Inc.
Javier Jaramillo, Schneider Automation Inc.
Denis Jean, 3COM Corporation
Kathy Jenczewski, Syscon-Micro Screen
Richard Kennady, Bahiatech Bahia Technologia Ltda
John Knowles, DEK Printing Machines Ltd.
Richard Lieske, DEK USA Inc.
Andy C. Mackie, Praxair Inc.
Nicholas C. Mescia, General Dynamics - ATS
Christine A. Miller, FORE Systems Inc.
Steve M. Moore, SMT
John Oliver, Acumen Technology
Ahne Oosterhof, A-Laser, Inc.
Deepak K. Pai, C.I.D., General Dynamics Information Sys. Inc.
Timothy M. Pitsch, Plexus Corp.
Jim R. Reed, Raytheon Systems Company
Joseph Renda, Speedline Technologies
Robert Rowland, RadiSys Corporation
Steve Sangillo, DMC Manufacturing, Inc.
Mark A. Schwarz, Vitronics Soltec
Dr. R. Sengupta, Electronics Regional Test Lab
Hans L. Shin, Pacific Testing Laboratories
Jan-Christer Stalnert, Ericsson Mobile Communications
Daan Terstegge, Signaal Communications
Richard Thompson, Ventura Electronics Assembly
David P. Torp, Kester Solder Division
Nancy Trumbull, TRMI, Inc.
Raymond J. Turner, Cabletron Systems Inc.
Rongxiang (Davis) Yang, Huawei Technologies Co., Ltd.
Michael W. Yuen, Compaq

# Table of Contents



## Figures



## Tables

This Page Intentionally Left Blank

# Stencil Design Guidelines

## 1 PURPOSE

This document provides guides for the design and fabrication of stencils for solder paste and surface-mount adhesive. It is intended as a guideline only.

**1.1 Terms and Definitions** All terms and definitions used throughout this handbook are in compliance with IPC-T-50. Definitions denoted with an asterisk (*) below are reprints from IPC-T-50. Other specific terms and definitions, essential for the discussion of the subject, are provided below.

**1.1.1 Aperture** An opening in the stencil foil.

**1.1.2 Aspect Ratio and Area Ratio**

Aspect Ratio = Width of Aperture / Thickness of Stencil Foil

Area Ratio = Area of Aperture Opening /Area of Aperture Walls

**1.1.3 Border** Peripheral tensioned mesh, either polyester or stainless steel, which keeps the stencil foil flat and taut. The border connects the foil to the frame.

**1.1.4 Contained Paste Transfer Head** A stencil printer head that holds, in a single replaceable component, the squeegee blades and a pressurized chamber filled with solder paste.

**1.1.5 Etch Factor** Etch Factor = Etched Depth / Lateral Etch in a chemical etching process

**1.1.6 Fiducials** Reference marks on the stencil foil (and other board layers) for aligning the board and the stencil when using a vision system in a printer.

**1.1.7 Fine-Pitch BGA/Chip Scale Package (CSP)** Ball grid array with less than 1 mm [39 mil] pitch. This is also known as Chip Scale Package (CSP) when the package size is no more than 1.2X the area of the original die size.

**1.1.8 Fine-Pitch Technology (FPT)*** A surface-mount assembly technology with component terminations on centers less than or equal to 0.625 mm [24.61 mil].

**1.1.9 Foil** The sheet used to create the stencil.

**1.1.10 Frame** A device onto which the foil is mounted. This may be tubular or cast aluminum with the border permanently mounted using an adhesive. Some foils can be mounted into a tensioning master case and do not require a border or a permanent fixturing of the foil to the frame.

**1.1.11 Intrusive Soldering** Intrusive soldering may also be known as paste-in-hole, pin-in-hole, or pin-in-paste soldering. This is a process in which the solder paste for the through-hole components is applied using the stencil, the through-hole components are inserted and reflow-soldered together with the surface-mount components.

**1.1.12 Modification** The process of changing an aperture in size or shape.

**1.1.13 Overprinting** The use of stencils with apertures larger than the pads or annular rings on the board.

**1.1.14 Pad** Metallized shape on the circuit board to which the terminal of a surface mount component is electrically or mechanically attached.

**1.1.15 Squeegee** A metal or rubber blade used to wipe across the stencil to force solder paste into openings in the stencil. Normally, squeegee is mounted at an angle such that the printing edge of the squeegee trails behind the print head and the face of the squeegee slopes forward.

**1.1.16 Standard BGA** Ball grid array with 1 mm [39 mil] pitch or larger.

**1.1.17 Stencil** A tool which may consist of a frame, border, and foil containing apertures through which solder paste, adhesive, or other media is transferred.

**1.1.18 Step Stencil** A stencil with more than one foil thickness level.

**1.1.19 Surface-Mount Technology (SMT)*** The electrical connection of components to the surface of a conductive pattern that does not utilize component holes.

**1.1.20 Through-Hole Technology (THT)*** The electrical connection of components to a conductive pattern by the use of component holes.

**1.1.21 Ultra-Fine Pitch Technology** A surface-mount assembly technology with component terminations on centers less than or equal to 0.40 mm [15.7 mil]

## 2 APPLICABLE DOCUMENTS

### 2.1 IPC[1]

**IPC-T-50** Terms and Definitions for Interconnecting and Packaging Electronic Circuits

**IPC-A-610** Acceptability of Electronic Assemblies

**IPC-SM-782** Surface Mount Design and Land Pattern Standard

**IPC-2511** Generic Requirements for Implementation of Product Manufacturing Description Data and Transfer Methodology

**IPC-7095** Design and Assembly Process Implementation of BGAs

### 2.2 Joint Industry Standard[1]

**J-STD-005** Requirements for Soldering Pastes

### 2.3 Barco/ETS[2]

**Gerber RS-274D** Format Reference Guide, Part Number 414-100-002

**Gerber RS-274X** Format User's Guide, Part Number 414-100-014

## 3 STENCIL DESIGN

### 3.1 Stencil Data

**3.1.1 Data Format** Regardless of the stencil fabrication method used, Gerber® data is the preferred data format. Possible alternative formats are GenCAM®[3], DXF, HP-GL, Barco, etc; however, they may need to be converted to Gerber® format prior to the stencil manufacturing process.

Gerber® data describes the file format that provides a language for communicating with the photo plotting system to produce a tool for chemically etched stencils. It is also used to produce the laser cut or electroformed stencils. While the actual data format may vary from file to file depending on the software package or designer, the data format commonly used by photo plotter and laser equipment is known as Gerber®.

**3.1.2 Gerber® Format** There are two standard Gerber® formats available:

- RS-274D - requires a data file listing the X-Y coordinates on the stencil where apertures are to be placed and formed, and a separate Gerber® aperture list that describes the size and shape of the various Gerber® apertures used to prepare the image.
- RS-274X - in this format the Gerber® aperture list is embedded in the data file.

**3.1.3 Aperture List** The aperture list is an ASCII text file containing D codes that define the size and shape for all apertures used within the Gerber® file. Without the aperture list, the software and photo plotting system cannot read the Gerber® data. Only the X-Y coordinates would be available with no size and shape data.

**3.1.4 Solder Paste Layer** The solder paste layer data is necessary to produce a stencil. If fiducials are required on the stencil, they should also be included in the solder paste layer.

**3.1.5 Data Transfer** Data can be transmitted to the stencil supplier via modem, FTP (file transfer protocol), e-mail attachment or disk. To ensure data integrity after transmitting and due to the large size of data files, it is suggested that the files be compressed prior to sending data. It is recommended that the full data file (the solder paste, solder mask, silk screen and copper layers) sent to the printed circuit board manufacturer be supplied to the stencil manufacturer. This allows the stencil manufacturer to optimize or make recommendations on aperture sizes based on actual pad sizes for the SMT land.

**3.1.6 Panelized Stencils** In those cases where it is desired to have more than one image on the stencil, the stencil patterns will be panelized and included in the data file. In those instances where the data file does not already contain the panelized stencil design, a readme file, panel drawing or order information must specify the location of the two or more designs. This could be a reference from the edge of the frame, distances between patterns, etc.

**3.1.7 Step-and-Repeat** In those cases where more than one image of the same design is to be printed, the data file for stencil fabrication should contain the stencil design in the step-and-repeat array. In those instances where the data file does not contain the step-and-repeat pattern, a readme file, panel drawing, or order information should specify:

- Total number of steps for the final array
- Number of steps in the X-direction along with dimensions from a specific feature to corresponding feature (such as fiducials, component pad locations, etc.).
- Number of steps in the Y-direction along with dimensions from a specific feature to corresponding feature (such as fiducials, component pad locations, etc.).

1. IPC, 2215 Sanders Road, Northbrook, IL 60062
2. Barco/ETS in North America, Telephone 860-291-7000, Fax 860-291-7021. For Europe and Asia, Telephone +32 9 21 69 441, Fax +32 9 21 69 870, Email etsinfo@barco.com
3. IPC Generic Computer Aided Manufacturing (GenCAM®) format. More information available at the back of document. Refer to IPC-2511 for details.

**3.1.8 Image Orientation/Rotation** In those cases where image orientation is not parallel to the frame or the step-and-repeat is not recti-linear (one or more images is rotated), the data for stencil fabrication should contain the oriented image. In those cases where it does not, a readme file, panel drawing or order information should specify this information (X- and Y-offsets) referencing stencil features.

**3.1.9 Image Location** To accommodate specific printers, the stencil image may have to be located in different positions inside the frame:

(a) center image

(b) center board/panel – requires board/panel outlines

(c) offset board/panel – requires board/panel outlines and reference locations

In those cases where this data is not included in the Gerber® data, a readme file, panel drawing or order information should specify this information referencing stencil features.

**3.1.10 Identification** Stencil should contain identification information such as part number, revision number, thickness, supplier's name and control number, date and method of manufacture.

**3.2 Aperture Design** A general aperture design guideline for various SMT components is shown in Table 1. Some of the factors influencing stencil aperture design are: component type, pad footprint, solder mask opening, board finish, aspect/area ratio, solder paste type and user process requirement.

**3.2.1 Aperture Size** The volume of solder paste applied to the board is mainly determined by the aperture size and foil thickness. Solder paste fills the stencil aperture during the squeegee cycle of the print operation. The paste should completely release to the pads on the board during the board/stencil separation cycle of the print operation. From the stencil viewpoint, the ability of the paste to release from the inner aperture walls to the board pad depends primarily on three major factors:

(1) the area and aspect ratios for the aperture design (See 3.2.1.1.)

(2) the aperture side wall geometry (See Section 4.4)

(3) the aperture wall finish (See Section 4.4)

**3.2.1.1 Area Ratio/Aspect Ratio** Both area ratio and aspect ratio are illustrated in Figure 1. A general design guide for acceptable paste release is >1.5 for aspect ratio and >0.66 for area ratio. The aspect ratio is a one-dimensional simplification of the area ratio. When the length is much greater than the width, the area ratio $\left(\frac{W}{2T}\right)$ reduces to a factor of the aspect ratio $\left(\frac{W}{T}\right)$.

When the stencil separates from the board, paste release encounters a competing process: solder paste will either transfer to the pad on the board or stick to the aperture side walls. When the pad area is greater than 0.66 of the inside aperture wall area, a complete paste transfer should occur.

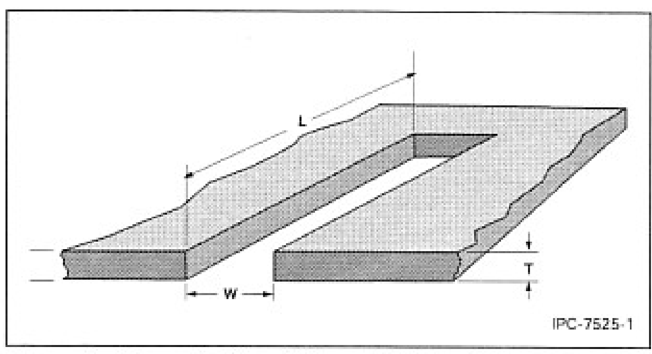



**Figure 1 Cross Sectional View of A Stencil Aperture**

$$\text{Aspect Ratio} = \frac{\text{Width of Aperture}}{\text{Thickness of Stencil Foil}} = \frac{W}{T}$$

$$\text{Area Ratio} = \frac{\text{Area of Pad}}{\text{Area of Aperture Walls}} = \frac{L \times W}{2 \times (L + W) \times T}$$

**3.2.2 Aperture Size Versus Board Pad Size** As a general design guide, the aperture size should be reduced compared to the board pad size. The stencil aperture is commonly modified with respect to the original pad design. Reductions in the area or changes in aperture shape are often desirable to enhance the processes of printing, reflow, or stencil cleaning. For instance, reducing the aperture size will decrease the possibility of stencil aperture to board pad misalignment. This reduces the chance for solder paste to be printed off the pad, which may lead to solder balls or solder bridging. Having a radiused corner for all apertures can promote stencil cleaning.

**3.2.2.1 Leaded SMD's** For leaded SMD's, e.g., J-leaded or gull-wing components with 1.3 - 0.4 mm [51.2 - 15.7 mil] pitch, the reduction is typically 0.03 - 0.08 mm [1.2 - 3.1 mil] in width and 0.05 - 0.13 mm [2.0 - 5.1 mil] in length.

**3.2.2.2 Plastic BGA's** Reduce circular aperture diameter by 0.05 mm [2.0 mil].

**3.2.2.3 Ceramic BGA's** Increase circular aperture dimension by 0.05 - 0.08 mm [2.0 - 3.1 mil] when this does not interfere with the solder mask and/or increase the stencil foil thickness to 0.2 mm [7.9 mil] and go one to one with the board pad. Refer to IPC-7095 for solder paste volume requirements.

**3.2.2.4 Fine-Pitch BGA and CSP** Square aperture with the width of the square equal to, or 0.025 mm [0.98 mil] less than, the diameter of the pad circle on the board. The

**Table 1 General Aperture Design Guidelines for Surface-Mount Devices**

| Part Type | Pitch | Pad Footprint Width | Pad Footprint Length | Aperture Width | Aperture Length | Stencil Foil Thickness Range | Aspect[2] Ratio Range | Area Ratio Range |
|---|---|---|---|---|---|---|---|---|
| PLCC | 1.25 mm [49.2 mil] | 0.65 mm [25.6 mil] | 2.00 mm [78.7 mil] | 0.60 mm [23.6 mil] | 1.95 mm [76.8 mil] | 0.15 - 0.25 mm [5.91 - 9.84 mil] | 2.3 - 3.8 | 0.88 - 1.48 |
| QFP | 0.65 mm [25.6 mil] | 0.35 mm [13.8 mil] | 1.50 mm [59.1 mil] | 0.30 mm [11.8 mil] | 1.45 mm [57.1 mil] | 0.15 - 0.175 mm [5.91 - 6.89 mil] | 1.7 - 2.0 | 0.71- 0.83 |
| QFP | 0.50 mm [19.7 mil] | 0.30 mm [11.8 mil] | 1.25 mm [49.2 mil] | 0.25 mm [9.84 mil] | [1.20 mm] 47.2 mil | 0.125 - 0.15 mm [4.92 - 5.91 mil] | 1.7 - 2.0 | 0.69 - 0.83 |
| QFP | 0.40 mm [15.7 mil] | 0.25 mm [9.84 mil] | 1.25 mm [49.2 mil] | 0.20 mm [7.87 mil] | [1.20 mm] 47.2 mil | 0.10 - 0.125 mm [3.94 - 4.92 mil] | 1.6 - 2.0 | 0.68 - 0.86 |
| QFP | 0.30 mm [11.8 mil] | 0.20 mm [7.87 mil] | 1.00 mm [39.4 mil] | 0.15 mm [5.91 mil] | 0.95 mm [37.4 mil] | 0.075 - 0.125 mm [2.95 - 3.94 mil] | 1.5 - 2.0 | 0.65 - 0.86 |
| 0402 | N/A | 0.50 mm [19.7 mil] | 0.65 mm [25.6 mil] | 0.45 mm [17.7 mil] | 0.60 mm [23.6 mil] | 0.125 - 0.15 mm [4.92 - 5.91 mil] | N/A | 0.84 - 1.00 |
| 0201 | N/A | 0.25 mm [9.84 mil] | 0.40 mm [15.7 mil] | 0.23 mm [9.06 mil] | 0.35 mm [13.8 mil] | 0.075 - 0.125 mm [2.95 - 3.94 mil] | N/A | 0.66 - 0.89 |
| BGA | 1.25 mm [49.2 mil] | CIR 0.80 mm [31.5 mil] | CIR 0.80 mm [31.5 mil] | CIR 0.75 mm [29.5 mil] | CIR 0.75 mm [29.5 mil] | 0.15 - 0.20 mm [5.91 - 7.87 mil] | N/A | 0.93 - 1.25 |
| Fine-Pitch BGA[1] | 1.00 mm [39.4 mil] | CIR 0.38 mm [15.0 mil] | CIR 0.38 mm [15.0 mil] | SQ 0.35 mm [13.8 mil] | SQ 0.35 mm [13.8 mil] | 0.115 - 0.135 mm [4.53 -5.31 mil] | N/A | 0.67 - 0.78 |
| Fine-Pitch BGA[1] | 0.50 mm [19.7 mil] | CIR 0.30 mm [11.8 mil] | CIR 0.30 mm [11.8 mil] | SQ 0.28 mm [11.0 mil] | SQ 0.28 mm [11.0 mil] | 0.075 - 0.125 mm [2.95 - 3.94 mil] | N/A | 0.69 - 0.92 |

**Note:**
1. It is assumed that the fine-pitch BGA pads are not solder mask defined.
2. N/A implies that only the area ratio should be considered.

square should have rounded corners. A guideline is 0.06 mm [2.4 mil] radiused corners for a 0.25 mm [9.8 mil] square and 0.09 [3.5 mil] corners for a 0.35 mm [14 mil] square.

**3.2.2.5 Chip Components - Resistors and Capacitors** Several aperture geometries are effective in reducing the occurrence of solder balls. All these designs are aimed at reducing excess solder paste trapped under the chip component. The most popular designs are shown in Figure 2, 3 and 4. These designs are commonly used for no-clean processes.

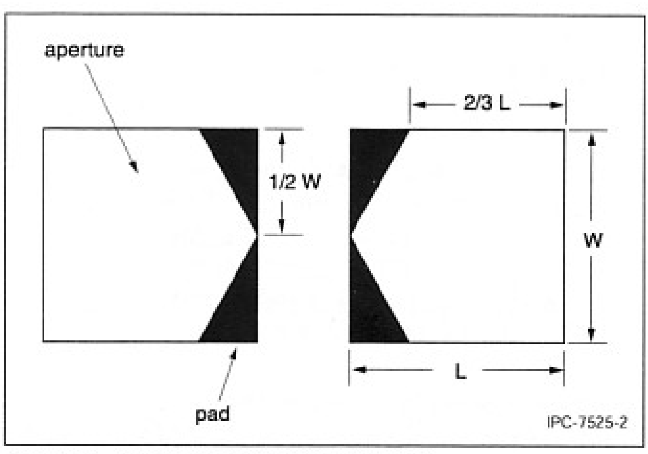



**Figure 2 Home Plate Aperture Design**

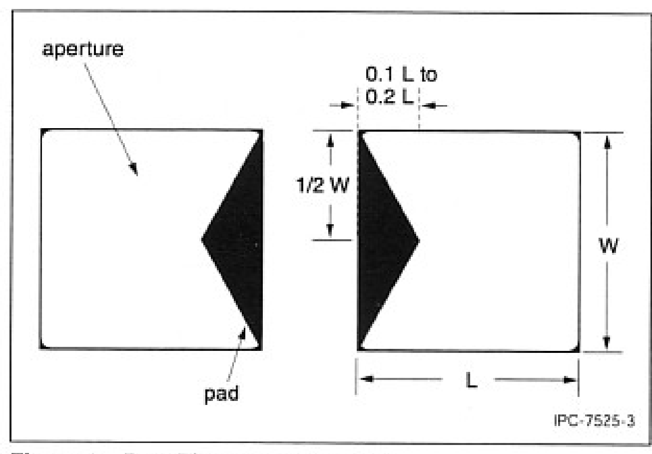



**Figure 3 Bow Tie Aperture Design**

**3.2.2.6 MELF, Mini-MELF Components** For MELF and Mini-MELF components, "C" shaped apertures are suggested. (See Figure 5). Dimensions of these apertures should be designed to match the geometry of component terminals.

**3.2.3 Glue Aperture Chip Component** The glue stencil is typically 0.15 - 0.2 mm [5.9 - 7.9 mil] thick. The glue aperture is placed in the center of the component solder pads. It is 1/3 the spacing between pads and 110% of the component width. (See Figure 6.)

More information about glue stencil will be made available in the next revision of this document.

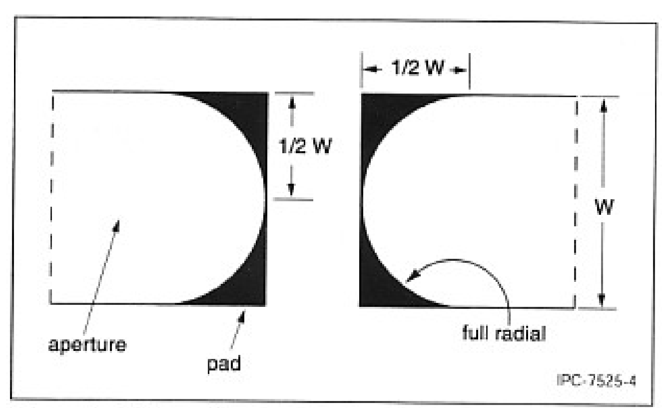



Figure 4 Oblong Aperture Design

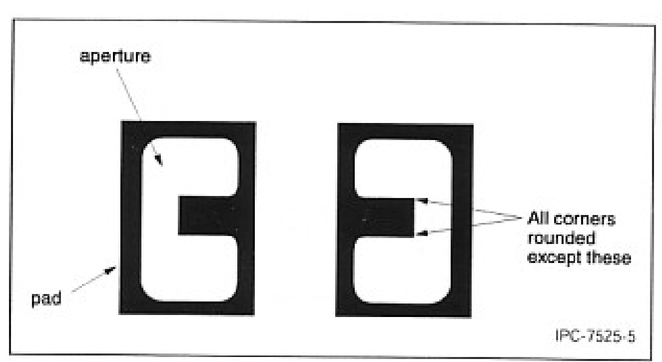



Figure 5 Aperture Design for MELF Components

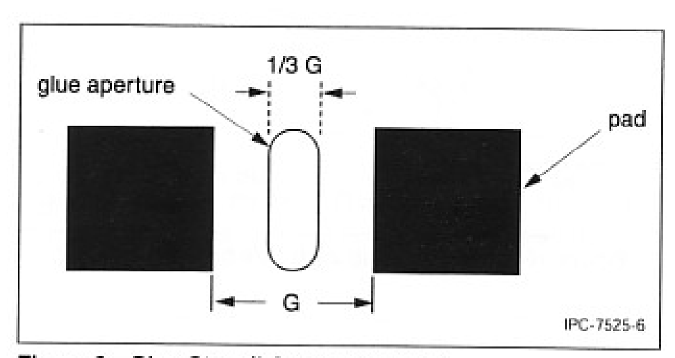



Figure 6 Glue Stencil Aperture Design

**3.3 Mixed Technology Surface-Mount/Through-Hole (Intrusive Reflow)** It is desirable to have a process where SMT and THT devices can both be:

(1) provided with printed solder paste

(2) placed on or in the board

(3) reflowed together.

The objective of stencil printing of solder paste for the intrusive reflow process is to provide enough solder volume after reflow to fill the hole and create acceptable solder fillets around the pins. Table 2 shows process window for a typical intrusive soldering process.

**3.3.1 Solder Paste Volume** A simple equation listed below describes the volume of solder paste required as shown in Figure 7.

$$V = T_S (L_O \times W_O)$$
$$= \frac{1}{S} \{T_B (A_H - A_P) + (F_T + F_B) + V_P\} - V_H$$

**Table 2 Process Window for Intrusive Soldering**

| | Maximum Limits | Desirable |
|---|---|---|
| Hole Diameter | 0.65 - 1.60 mm [25.6 - 63.0 mil] | 0.75 - 1.25 mm [29.5 - 49.2 mil] |
| Lead Diameter | Hole diameter minus 0.075 mm [2.95 mil] | Hole diameter minus 0.125 mm [4.92 mil] |
| Paste Overprinting | 6.35 mm [250 mil] | <4.0 mm [157 mil] |
| Stencil Foil Thickness | 0.125 - 0.635 mm [4.92 - 25.0 mil] | 0.15 mm [5.91 mil], 0.20 mm [7.87 mil] for fine-pitch |

Where:

$V$ is volume of solder paste required
$V_P$ is the solder volume left on the top and/or bottom board pad
$S$ is the solder paste shrink factor
$A_H$ is the cross sectional area of the through-hole
$A_P$ is the cross sectional area of the through-hole pin
$T_B$ is the thickness of the board
$F_T + F_B$ is the total fillet volume required
$T_S$ is the thickness of the stencil foil
$L_O$ is the length of the overprint aperture
$L_P$ is the length of the pad
$W_O$ is the width of the overprint aperture
$W_P$ is the width of the pad
$V_H$ is solder paste filling the hole during the printing operation

It is desirable to keep the copper pad around the through-hole as small as possible. It is also desirable to keep the clearance between the pin and the through-hole and the length of the pin as small as possible. By doing this less solder paste volume will be required.

**Note:** Solder paste volume filling the hole can vary from 0% to 100% depending on the print setup. Contained paste transfer heads are effective in achieving close to 100% while metal squeegee blades with a high attack angle and high print speed will deliver minimum paste into the hole.

Following are three stencil designs used to deliver the through-hole solder paste:

(1) Non-step stencil

(2) Step stencil

(3) Two-print stencil

**3.3.1.1 Overprint Without Step** This is the stencil of choice when it can deliver enough solder paste to satisfy the through-hole requirement. A cross section of this type stencil is shown in Figure 8.

An example of when this stencil could be used is a two-row connector on 2.5 mm [98.4 mil] pitch with 1.1 mm [43.3 mil] diameter though-holes, 0.9 mm [35.4 mil] diameter pins, 1.2 mm [47.2 mil] thick board and no other components within 3.8 mm [150 mil] of the through-hole openings. An overprint stencil aperture of 2.2 mm [86.6 mil]

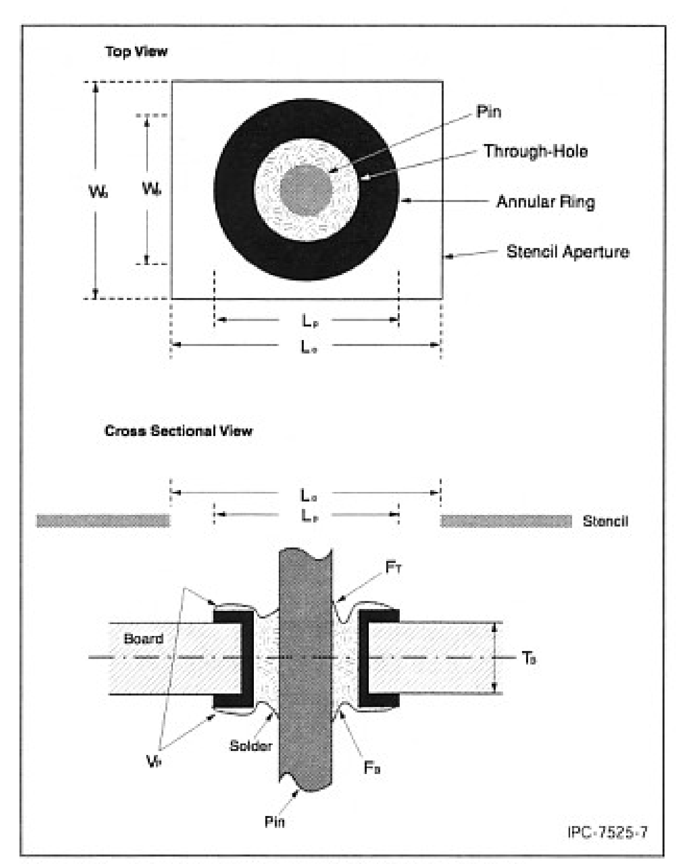



**Figure 7 Through-Hole Solder Paste Volume**

wide and 5.1 mm [200 mil] long with a stencil foil thickness of 0.15 mm [5.91 mil] can deliver sufficient solder paste.

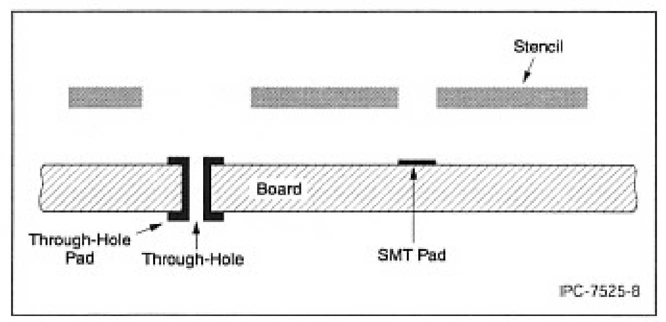



**Figure 8 Overprint without Step**

**3.3.1.2 Overprint With Step** If the board is thicker, the hole is bigger, or the pin is smaller, more solder paste volume will be required. In this case, a step stencil may be needed to provide sufficient solder paste volume for the THT parts without providing too much paste on the SMT pads. An example of this type stencil is shown in Figure 9.

K1 and K2 are keep-out distances. K2 is the distance between the through-hole aperture and the step edge. As a general design guide K2 can be as low as 0.65 mm [25.6 mil]. K1 is the distance from the step edge to the nearest aperture in the step-down area. As a general design guide, K1 should be 0.9 mm [35.4 mil] for every 0.025 mm [0.98 mil] of step-down thickness. As a simple guideline, K1

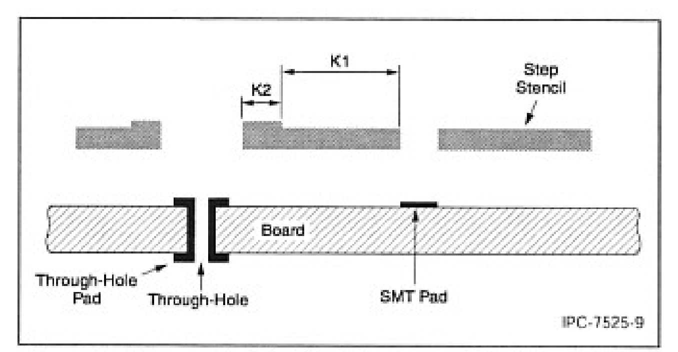



**Figure 9 Overprint with Step (Squeegee Side)**

should be 36x the step-down thickness. For example, a 0.2 mm [7.9 mil] stencil foil with a step down to 0.15 mm [5.9 mil] would require a K1 keep-out distance of 1.8 mm [70.9 mil]. It is also possible to put the step on the contact side of the stencil instead of the squeegee side. This is shown in Figure 10. This type of step is sometimes more convenient when using metal squeegee blades and is highly recommended for contained paste transfer heads. The same keep-out rules apply.

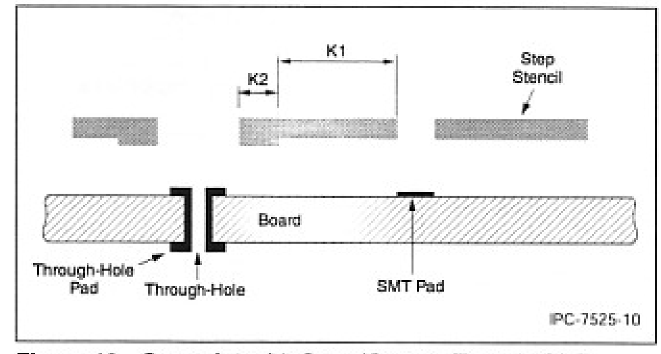



**Figure 10 Overprint with Step (Contact/Board Side)**

**3.3.1.3 Two-Print Stencil** Some through-hole devices have small pins with large holes or dense spacing with thick boards. In either case insufficient solder paste volume is deliverable using the first two stencil designs. The two-print stencil can deliver large amounts of solder paste into the through-holes. In this design, a normal surface-mount stencil, typically 0.15 mm [5.9 mil] thick, is used to print the surface-mount solder bricks. While the surface-mount paste is still tacky, a thick stencil is used to print the through-hole paste. Normally this requires a second stencil printer set up in line to perform this printing. This stencil can be as thick as required. However, 0.4 to 0.75 mm [16 to 30 mil] is typical. When stencil foil thickness requirements exceed 0.5 mm [20 mil], laser cut electropolished apertures provide better paste release and overall print performance due to the excellent wall geometry. The contact side of this stencil foil is relief etched at least 0.25 mm [9.84 mil] deep in any area where surface-mount bricks have been previously printed. A cross section of the two-print through-hole stencil is shown in Figure 11.

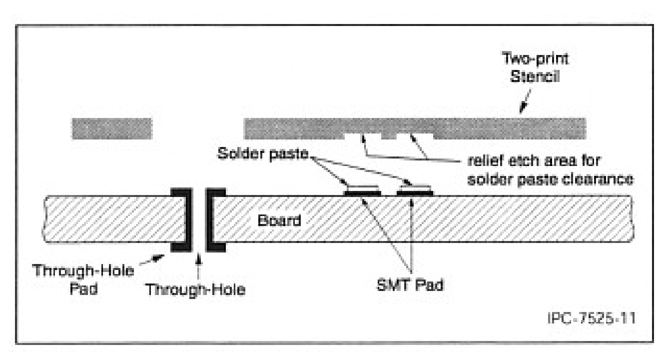



**Figure 11 Two-Print Through-Hole Stencil**

**3.4 Mixed Technology Surface-Mount/Flip Chip** A sample application for this technology is a PCMCIA card having flip chips, TSOPs, and chip components. It is desirable to place flip chip and SMT components on the card and reflow all the components simultaneously.

**3.4.1 Two-Print Stencil for Surface-Mount/Flip Chip** The two-print stencil configuration can perform this task. The first step in this process is to print flip chip solder paste or flip chip flux on the board flip chip pad sites. A stencil to do this is normally 0.05 or 0.075 mm [2.0 or 3.0 mil] thick with 0.13 to 0.18 mm [5.12 to 7.09 mil] apertures. While the flip chip paste/flux is still tacky, the surface-mount stencil is used to print the surface-mount paste bricks. An example of this stencil would be one that is 0.18 mm [7.09 mil] thick with a relief etch of 0.10 mm [3.93 mil] in the area of the flip chip paste/flux. An example of a two-print stencil for this application is shown in Figure 12.

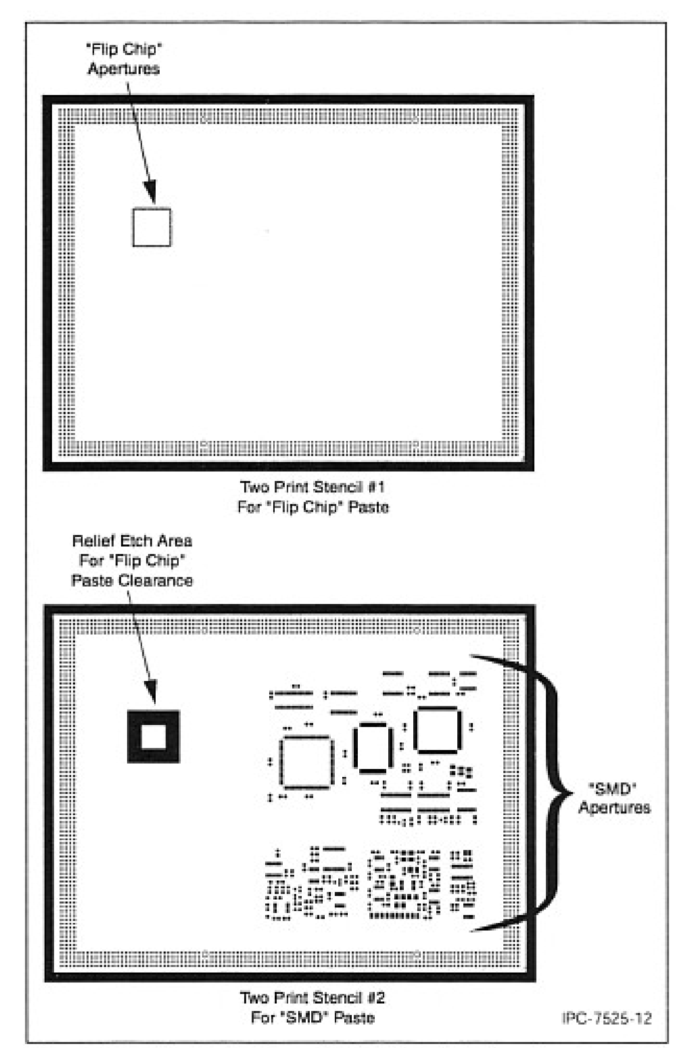



**Figure 12 Two-Print Stencil for Mixed Technology**

**3.5 Step Stencil Design** There are several applications where a stencil with multiple foil thicknesses may be desirable. These designs are outlined below.

**3.5.1 Step-Down Stencil** This type of stencil is useful when it is desirable to print fine-pitch devices using a thinner stencil foil but print other devices using a thicker stencil foil. For example there may be a fine-pitch BGA of 0.5 mm [20 mil] pitch that requires a 0.1 mm [3.9 mil] foil thickness to achieve an area ratio of greater than 0.66 but at the same time there are other devices on the same board that need a thickness of 0.13 to 0.15 mm [5.1 to 5.9 mil]. The stencil design would have a step area at 0.1 mm [3.9 mil] thick in the fine-pitch BGA portion while the remainder of the stencil foil is 0.15 mm [5.9 mil] thick. The step can be on the squeegee side or on the contact side. See 3.3.1.2 for keep-out design guidelines.

**3.5.2 Step-Up Stencil** This type stencil is useful when it is desirable to print thicker solder paste in a small portion of the stencil. An example would be a ceramic BGA where it is necessary to get 0.2 mm [7.9 mil] paste height because of ball coplanarity but 0.15 mm [5.9 mil] height on all other surface-mount component pads. In this case the stencil foil is stepped up from 0.15 mm [5.9 mil] to 0.2 mm [7.9 mil] in the area of the ceramic BGA. Another example is a through-hole edge connector that requires additional solder paste volume. In this case the stencil foil may be 0.15 mm [5.9 mil] thick everywhere except in the area of the edge connector where the stencil foil may be 0.3 mm [12 mil] thick.

**3.5.3 Step Stencil for Contained Paste Transfer Heads** As a general design guide, step should not exceed 0.05 mm [2.0 mil].

**3.5.4 Relief-Etch Stencil** This type of stencil has relief step pockets on the contact/board side of the stencil foil. There are several applications where relief-etch stencils are useful. Some examples are:

- Relief pocket for a bar code label on the board. The stencil foil might be 0.15 mm [5.9 mil] thick with a 0.08 mm [3.1 mil] relief for the bar code label.
- Test via relief pockets. The stencil foil has relief pockets over each raised test via to allow the stencil foil to sit flat and gasket to the board.

- Two-print stencil. This stencil foil has deep relief pockets in the areas where surface-mount solder paste was previously printed. (See 3.3.1.3 and 3.4.1.) An example of this stencil would be a stencil 0.5 mm [20 mil] thick for printing paste in and around through-hole with relief step on the contact side 0.3 mm [12 mil] thick to clear surface-mount solder paste previously printed.
- Use of solder resist pedestals at the corners of ceramic components. A relief etch on the stencil foil provides good gasketing. The standoff of ceramic leadless components can be increased to accommodate cleaning under the component and to increase the length of solder joint.

**3.6 Fiducials** Depending on the vision location, fiducials are located on the squeegee or contact side and filled with black epoxy for contrast. Typically, they are solid, round dots 1.0 to 1.5 mm [39.4 to 59.1 mil] in diameter. They may be half-etched, laser engraved or etched through the entire stencil.

**3.6.1 Global Fiducials** Fiducials that are placed a minimum of 5 mm [0.20 in] from the board corners in three locations.

**3.6.2 Local Fiducials** Fiducials that are placed near critical components, e.g., fine-pitch QFP.

## 4 STENCIL FABRICATION

**4.1 Foils** Stainless steel is the preferred metal for chemical etch and laser cut technology. Other metals, as well as plastics, may be specified. For electroform technology, a hard nickel alloy is preferred.

**4.2 Frames** Refer to the OEMs stencil printer operation manual for available frame sizes. Frames may be tubular or cast aluminum with the border permanently mounted using an adhesive. Some foils can be mounted into a tensioning master case and do not require a border or a permanent fixturing of the foil to the frame.

**4.3 Stencil Border** Polyester is the standard material; stainless steel is optional.

**4.4 Stencil Fabrication Technologies** The fabrication process for stencils may involve additive or subtractive methods. In additive processes such as electroforming, metal is added to form stencil foils. In subtractive processes, metal is removed from foils to create apertures. Laser cut and chemical etch are examples of subtractive processes.

**4.4.1 Chemical Etch** Chemically etched stencils are produced using photo-imageable resist laminated on both sides of metal foils cut to specific frame sizes. A double-sided phototool, held in precise alignment usually with registration pins, is used to expose the stencil aperture image onto the resist. Aperture images exposed on the resists are reduced in size compared with the desired aperture dimensions, accounted by an etch factor. The etch factor describes the amount of lateral etching that takes place as the chemical etches through the thickness of metal foil. The exposed resist is then developed, leaving bare metal where apertures are desired. The metal foil is etched from both sides in a liquid chemical, creating apertures as specified. The remaining resist is then stripped away and a stencil foil is produced.

**4.4.2 Laser Cut** Laser cut stencils are produced from data run by software of the laser equipment. Unlike chemically etched stencils, no phototool is required. Since stencils are cut from one side only, tapered aperture wall is an inherent part of laser cut stencils. Unless otherwise specified, apertures are larger on the contact side than on the squeegee side. (See 4.4.5)

**4.4.3 Electroform** Electroforming is an additive stencil fabrication method utilizing photo-imageable resist and an electroplating process. Photo-imageable resist is placed on a metal mandrel. Thickness of the resist is greater than the final stencil thickness desired. The apertures are imaged onto the resist and the resist is developed, leaving resist pillars where apertures are desired. The mandrel with resist pillars is placed in a nickel plating tank where nickel is electroplated onto the mandrel. When the desired stencil thickness is reached, the mandrel is removed from the plating tank. Lastly, the resist pillars are stripped and the nickel stencil foil is separated from the mandrel.

**4.4.4 Hybrid** Where a mixture of standard and fine-pitch assemblies is present on a board, the stencil fabrication process may be a combination of laser cut and chemical etch. The stencils produced are referred to as laser-chem combination or hybrid stencils.

**4.4.5 Trapezoidal Apertures** A trapezoidal aperture may be used to enhance solder paste release. In chemical etch processes, the trapezoidal dimension, Z, (see Figure 13) can be specified. For laser cut or electroform processes trapezoidal aperture is an inherent part of process. Stencil vendors should be contacted for dimensions.

**4.4.6 Additional Options** Further processing may be desired on certain methods of stencil fabrication to reduce friction between solder paste and side walls for improved paste release. Choices of further processing are:

- *Polishing:* A subtractive process. Either chemical polishing or electropolishing process.
- *Nickel plating:* An additive process.

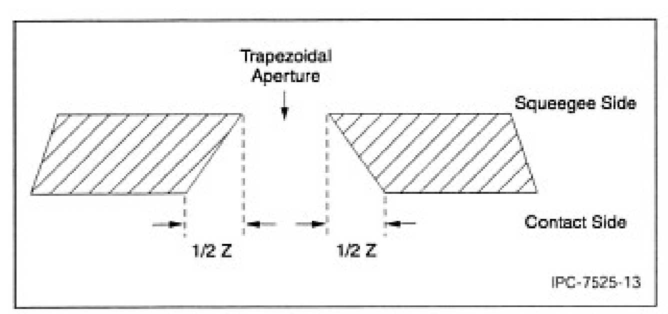



**Figure 13 Trapezoidal Apertures**

## 5 STENCIL MOUNTING

**5.1 Location of Image on Metal** The board is centered or offset on the metal. Board corner marks or the board outline is used for location. If these were not present on the data, the actual board image or global fiducials would be used for alignment. Where more than one boards or panels are placed on one stencil, a minimum of 50 mm [2.0 in] is recommended between the images.

**5.2 Centering** It is recommended that the stencil be centered on the frame for most uniform mechanical tensioning and print results. The image can be offset to meet specific requirements of the stencil printer.

**5.3 Additional Design Guidelines** Unless otherwise specified, additional design guidelines are:

- Minimum 20 mm [0.79 in] border is recommended from the edge of frame to the edge of metal.
- Minimum 50 mm [2.0 in] from the inside edge of glue border to the edge of image is suggested for solder paste storage and squeegee travel.

## 6 STENCIL ORDERING

Stencil information is typically communicated between the user and supplier through an order form (or checklist) created by the supplier. File data, types of material, fabrication methods, and special requests are examples of what is typically included on an order form (see Appendix A).

## 7 USER STENCIL INSPECTION/VERIFICATION

After receiving a stencil from the supplier, users are recommended to generate an inspection checklist to verify that the stencil has been fabricated correctly and that no damage has occurred during the shipping process. The following items can be used as a guideline for inspecting an incoming stencil:

- The foil should be inspected for chemical corrosion.
- The foil should be inspected for handling damage (e.g., dents, creases, metal voids).
- Tension of border should be checked.
- Correct spacing between the image and the frame should be verified (based on printer manufacturer's specification). A printed wiring board or transparent image of the board (e.g., mylar film with backlighting equipment) should be held up against the image of stencil to check for correct spacing between the board and the edge of stencil frame.
- The image of stencil should be observed to match the image of board. Modification of aperture size and/or shape should be verified.
- Border should be checked for proper adhesion to the foil and for any handling damage.
- The size and type of frame should be checked.
- Correct part number and revision number, etched or engraved in the stencil should be verified.
- Foil thickness should be verified.
- For a step stencil, the correct step level should be verified.
- Fiducial quality and location (correct side of foil) should be inspected.

## 8 STENCIL CLEANING

Proper setup and cleaning of stencil helps ensure continued repeatable printing performance. Cleaning processes need to be compatible with materials used in the manufacture of stencils. Paste or adhesive manufacturers, stencil manufacturers, and cleaning equipment manufacturers should be consulted as life of stencils, integrity of fiducials, and quality of glue bead may be affected.

## 9 END OF LIFE

Stencils should be inspected periodically for damage that would contribute to decreased printing performance. Refer to Section 7 for inspection guidelines.

# APPENDIX A: EXAMPLE ORDER FORM

Customer / Contact Name: ______________________________

Ship To: ______________________ Bill To: ______________________

Shipping Method: ______________________ Due Date: ______________________

Stencil Part Number and Revision Number: ______________________
(to be half-etched or engraved on squeegee / contact side of stencil)

File(s) Name: ______________________ File Type: ______________________

Additional File Information: ______________________ (e.g., Data Format, Coordinates, Units)

Data Transmitted By: [ ] Modem [ ] Email [ ] FTP [ ] Disk

Stencil Fabrication Method: [ ] Chemical Etch [ ] Laser Cut [ ] Laser-Chem Hybrid
[ ] Electroform [ ] ______________

Frame Size: ______________________
- Provided By: [ ] Customer [ ] Stencil Supplier
- Frame Coated: [ ] Yes [ ] No
- Printer Type: ______________________

Metal Thickness: ______________________

Step-Down Thickness (Drawing Required for Step Locations): ______________________

Metal Type: [ ] Stainless Steel [ ] ______________

Location of Pattern on Stencil: [ ] Center Image [ ] Center Board [ ] Offset (Drawings Required)

Fiducials:
- [ ] None
- [ ] Half Etch / Engraved Contact Side
- [ ] Half Etch / Engraved Squeegee Side
- [ ] Full Etch / Cut Through, Not Filled
- [ ] Full Etch / Cut Through, Filled with Contrasting Epoxy

[ ] Panelization Provided By: [ ] Customer [ ] Stencil Supplier
Dimensions Required: ______________

[ ] Step-and-Repeat Provided By: [ ] Customer [ ] Stencil Supplier
Dimensions Required: ______________

Border: [ ] Polyester [ ] Stainless Steel

Polish: [ ] Yes [ ] No

Nickel Plate: [ ] Yes [ ] No

Special Modifications, Editing, or Instructions: ______________________

______________________________________________

______________________________________________

______________________________________________




# Standard Improvement Form

**IPC-7525**

The purpose of this form is to provide the Technical Committee of IPC with input from the industry regarding usage of the subject standard.

Individuals or companies are invited to submit comments to IPC. All comments will be collected and dispersed to the appropriate committee(s).

If you can provide input, please complete this form and return to:

IPC
2215 Sanders Road
Northbrook, IL 60062-6135
Fax 847 509.9798

1. I recommend changes to the following:

___ Requirement, paragraph number ________

___ Test Method number ________, paragraph number ________

The referenced paragraph number has proven to be:

___ Unclear ___ Too Rigid ___ In Error

___ Other ________________________________________

2. Recommendations for correction:

________________________________________

________________________________________

________________________________________

________________________________________

3. Other suggestions for document improvement:

________________________________________

________________________________________

________________________________________

________________________________________

Submitted by:

| | |
|---|---|
| Name | Telephone |
| Company | E-mail |
| Address | |
| City/State/Zip | Date |




# ANSI/IPC-T-50 Terms and Definitions for Interconnecting and Packaging Electronic Circuits Definition Submission/Approval Sheet

The purpose of this form is to keep current with terms routinely used in the industry and their definitions. Individuals or companies are invited to comment. Please complete this form and return to:

IPC
2215 Sanders Road
Northbrook, IL 60062-6135
Fax: 847 509.9798

SUBMITTOR INFORMATION:

Name: ______________________

Company: ______________________

City: ______________________

State/Zip: ______________________

Telephone: ______________________

Date: ______________________

❑ This is a **NEW** term and definition being submitted.
❑ This is an **ADDITION** to an existing term and definition(s).
❑ This is a **CHANGE** to an existing definition.

| Term | Definition |
|---|---|
| | |
| | |
| | |
| | |
| | |

If space not adequate, use reverse side or attach additional sheet(s).

Artwork: ❑ Not Applicable ❑ Required ❑ To be supplied
❑ Included: Electronic File Name: ______________________

Document(s) to which this term applies: ______________________

Committees affected by this term: ______________________

**Office Use**

| **IPC Office** | **Committee 2-30** |
|---|---|
| Date Received: ____________ | Date of Initial Review: ____________ |
| Comments Collated: ____________ | Comment Resolution: ____________ |
| Returned for Action: ____________ | Committee Action: ❑ Accepted ❑ Rejected |
| Revision Inclusion: ____________ | ❑ Accept Modify |

**IEC Classification**

Classification Code • Serial Number

Terms and Definition Committee Final Approval Authorization:

Committee 2-30 has approved the above term for release in the next revision.

Name: ____________ Committee: IPC 2-30 Date: ____________

## Why of GenCAM® Now

**GenCAM** promises to alleviate much of the pain that the industry has endured for the past decade in regards to the transfer of information from the designer to the manufacturer.

GenCAM, simply put, is a ASCII format, driven by domain experts that represent printed board fabrication and test; list of materials, assembly, inspection and in-circuit test; as well as, documentation, administration, and configuration control management issues. These experts have worked for two years to make the GenCAM standard the most robust data description format of its time. GenCAM is:

- User friendly, easy to understand by manufacturing personnel
- Segmented into twenty linked sections to eliminate the need for redundancy
- A single file, able to completely describe a printed board, a printed board assembly, an assembly array, multiple assemblies on a sub-panel, a board fabrication panel, quality assessment coupons, and assembly/test fixtures.
- Easy to implement into any design or manufacturing computer system.
- Keyword driven, using terminology familiar to industry experts.
- Manufacturing processes oriented, to enable future "plug & play technology.
- Fully supported as an ANSI standard for GenCAM 1.0 or 1.5 and future releases.
- Open to continuous review and improvement to facilitate technology innovations and software development diversity.
- Establishing a level playing field software implementation concept for all CAD/CAM tool developers.
- Syntax and semantic checkable through the use of compliance software.
- Able to be parsed into manufacturing entities while maintaining the complete assembly database.

All of the features of GenCAM have one major purpose, which is to provide intelligent information about the characteristics of an electronic assembly and its tooling, thus reducing the cycle time required to start producing the product.

Elimination of the ambiguities associated with communication between engineering and manufacturing will also reduce the scrap when product is produced that does not meet the customer expectations.

Thus there are many benefits for using the new ANSI approved IPC standard. Some of these are identified as:

### The User Benefits

Cycle time is improved as GenCAM reduces the need to spoon-feed the supply chain

Use of GenCAM improves supply chain management needed as more services are being out-sourced

Establishes a valuable archiving capability for fabrication and assembly tooling enhancement

Improves equipment re-procurement capability

Segmentation of the GenCAM file avoids need to distribute proprietary product performance data

Provides method for improvement feedback

Establishes a methodology for supply chain management and maintaining consistent revision level status of product being manufactured.

### The Designer Benefits

Ability to provide complete description of one or multiple electronic assemblies

Establishes the communication link between design and manufacturing disciplines

Facilitates the re-use of graphical data

Direct correlation with CAD library methodology

Allows the description of tolerance conditions for accept/reject criteria

Brings design into close contact with DFM issues

### Electronic Assembly Benefits

Complete integrated bill of material

Identifies component substitution allowances

Can accommodate several BOM configurations in a single file

Establish flexible reuse of component package data

Supports sub-panel or assembly array descriptions

Considers all electrical and mechanical component instances, including orientation, board mounting side




*For more information, please visit the GenCAM website at www.gencam.org or contact Deiter Bergman at bergdi@ipc.org*




ASSOCIATION CONNECTING
ELECTRONICS INDUSTRIES

ISBN #1-580982-45-X

2215 Sanders Road, Northbrook, IL 60062-6135
Tel. 847.509.9700 Fax 847.509.9798
www.ipc.org